

This model was tested in Cura 3.1.0. and printed on an Ultimaker 2 in PLA material.

To avoid printing problems, we recommend the following **settings**:

Quality

Layer Height: 0.1 mm

Initial Layer Height: 0.3 mm

Slicing Tolerance: Middle

Line Width: 0.4 mm

Wall Line Width: 0.4 mm

Outer Wall Line Width: 0.4 mm

Inner Wall(s) Line Width: 0.4 mm

Top/Bottom Line Width: 0.4 mm

Infill Line Width: 0.4 mm

Skirt/Brim Line Width: 0.4 mm

Support Line Width: 0.4 mm

Initial Layer Line Width: 100%

Shell

Wall Thickness: 0.8 mm

Wall Line Count: 2

Outer Wall Wipe Distance: 0.2 mm

Top Surface Skin Layers: 0

Top/Bottom Thickness: 0.8 mm

Top Thickness: 0.8 mm

Top Layers: 8

Bottom Thickness: 0.8 mm

Bottom Layers: 8

Top/Bottom Pattern: Lines

Bottom Pattern Initial Layer: Lines

Top/Bottom Line Directions: []

Outer Wall Inset: 0 mm

Compensate Wall Overlaps: Check

Compensate Outer Wall Overlaps: Check
Compensate Inner Wall Overlaps: Check
Fill Gaps Between Walls: Everywhere
Filter Out Tiny Gaps: Check
Horizontal Expansion: 0 mm
Initial Layer Horizontal Expansion: 0 mm
Z Seam Alignment: Sharpest Corner
Seam Corner Preference: Hide Seam
Ignore Small Z Gaps: Check
Extra Skin Wall Count: 1

Infill

Infill Density: 20%
Infill Line Distance: 4.0 mm
Infill Pattern: Grid
Infill Line Directions: []
Infill X Offset: 0 mm
Infill Y Offset: 0 mm
Infill Overlap Percentage: 10%
 Infill Overlap: 0.04 mm
Skin Overlap Percentage: 5%
 Skin Overlap: 0.02 mm
Infill Wipe Distance: 0.1 mm
Infill Layer Thickness: 0.1 mm
Gradual Infill Steps: 0
Infill Before Walls: Check
Minimum Infill Area: 0 mm²
Skin Removal Width: 0.8 mm
 Top Skin Removal Width: 0.8 mm
 Bottom Skin Removal Width: 0.8 mm
Skin Expand Distance: 0.8
 Top Skin Expand Distance: 0.8
 Bottom Skin Expand Distance: 0.8

Maximum Skin Angle for Expansion: 90°
Minimum Skin Width for Expansion: 0.0

Material

Enable Retraction: Check
Retraction Extra Prime Amount: 0 mm³
Retraction Minimum Travel: 0.8 mm
Maximum Retraction Count: 90
Minimum Extrusion Distance Window: 6.5 mm
Nozzle Switch Retraction Distance: 16 mm
Nozzle Switch Retraction Speed: 20 mm/s
Nozzle Switch Retract Speed: 20 mm/s
Nozzle Switch Prime Speed: 20 mm/s

Speed

Print Speed: 45 mm/s
Infill Speed: 50 mm/s
Wall Speed: 22.5 mm/s
Outer Wall Speed: 22.5 mm/s
Inner Wall Speed: 45 mm/s
Top/Bottom Speed: 15 mm/s
Support Speed: 45 mm/s
Support Infill Speed: 45 mm/s
Travel Speed: 80 mm/s
Initial Layer Speed: 15 mm/s
Initial Layer Print Speed: 15 mm/s
Initial Layer Travel Speed: 26.6667 mm/s
Skirt/Brim Speed: 15 mm/s
Maximum Z Speed: 0 mm/s
Number of Slower Layers: 2

Travel

Combing Mode: All
Avoid Printed Parts when Traveling: Check
Travel Avoid Distance: 0.625 mm
Cooling
Enable Print Cooling: Check
Fan Speed: 100%
 Regular Fan Speed: 100%
 Maximum Fan Speed: 100%
Regular/Maximum Fan Speed Threshold: 10 s
Initial Fan Speed: 0%
Regular Fan Speed at Height: 0.3 mm
 Regular Fan Speed at Layer: 2
Minimum Layer Time: 5 s
Minimum Speed: 10 mm/s

Support

Generate Support: Check
Support Placement: Everywhere
Support Overhang Angle: 60°
Support Pattern: Zig Zag
Connect Support ZigZags: Check
Support Density: 15 %
 Support Line Distance: 1.3333 mm
Support Z Distance: 0.3 mm
 Support Top Distance: 0.3 mm
 Support Bottom Distance: 0.3 mm
Support X/Y Distance: 0.7 mm
Support Distance Priority: Z overrides X/Y
Minimum Support X/Y Distance: 0.2 mm
Support Stair Step Height: 0.3 mm
Support Stair Step Maximum Width: 5.0 mm
Support Join Distance: 2.0 mm
Support Horizontal Expansion: 0.2 mm
Support Infill Layer Thickness: 0.1 mm

Use Towers: Check
Tower Diameter: 3.0 mm
Minimum Diameter: 3.0 mm
Tower Roof Angle: 65°

Build Plate Adhesion

Build Plate Adhesion Type: Brim
Skirt/Brim Minimum Length: 250 mm
Brim Width: 8.0 mm
Brim Line Count: 18
Brim Only on Outside: Check

Mesh Fixes

Union Overlapping Volumes: Check
Maximum Resolution: 0.01 mm
Merged Meshes Overlap: 0.15 mm

Special Modes

Print Sequence: All at Once
Surface Mode: Normal

Disclaimer: This model will look outstanding if printed on SLA/SLS 3D printer. The accuracy of the model printed on FFF printer can vary from the result shown in the pictures.